XYCENEWSNOTE

Xyce is a scalable, analog network (e.g., circuit) simulator developed by Sandia under ASC funding to support large-scale advanced circuit simulations and analysis for problems critical to Sandia's nuclear weapons mission.

New Xyce capability demonstrated: integrated electro-mechanical system simulation

Department 1437 - Electrical & Microsystems Modeling

Overview

In May 2006, the Xyce™ (http://www.cs.sandia.gov/Xyce) team at Sandia National Laboratories, led by Eric Keiter, further demonstrated the flexibility and power of the Xyce application code. Specifically, the team built upon the code's general integrated-network solution engine by incorporating a theoretical, yet realistic, compact-model of a Micro-Electro-Mechanical System (MEMS) switch into its model library and simulating its switching behavior as driven by a digital signal.

The demonstration switch model in Xyce, implemented by Richard Schiek, is based on a two-ODE (Ordinary Differential Equation) description of a MEMS switch that is intended for demonstration purposes only and does not represent any MEMS device currently being designed or built by Sandia. Instead, this model and its integration within Xyce demonstrates a "proof-of-principle" that will show the way forward for further R&D for Xyce and the development of partnerships between Centers 1400 and 1700. Overall, the goal is to provide a systems-level simulation capability representing the integrated electro-mechanical behavior for design and analysis that will have a real impact on MEMS designs at Sandia for critical national security missions.

The Xyce code has been under development for several years as a key ASC application for modeling analog circuits at high levels of integration and in

extreme environments (e.g., prompt radiation). This new ability to incorporate mechanical "devices" into its compact model library enables Sandia designers to take a larger view of their MEMS designs as they will be able to simulate multi-domain assemblies.

Switch Properties

The fictitious, two-state switch is modeled as a
- mass in a potential field
- mass experiences drag
- time dependent impulsive force moves mass

As mentioned, the individual switch model is constructed of two ODEs:

$$\frac{dx}{dt} = v$$

$$\frac{dv}{dt} = \frac{1}{m}\left(F_i(t) - \frac{dU}{dx} - \alpha \cdot v\right)$$

where $\alpha, m, U(x)$ are design parameters needing calibration.

The resulting simple switch model has the following characteristics:
- captures relevant physics (switching speed, oscillations, dynamic behavior, powerless state)
- allows fast simulation of the device under many driving conditions.
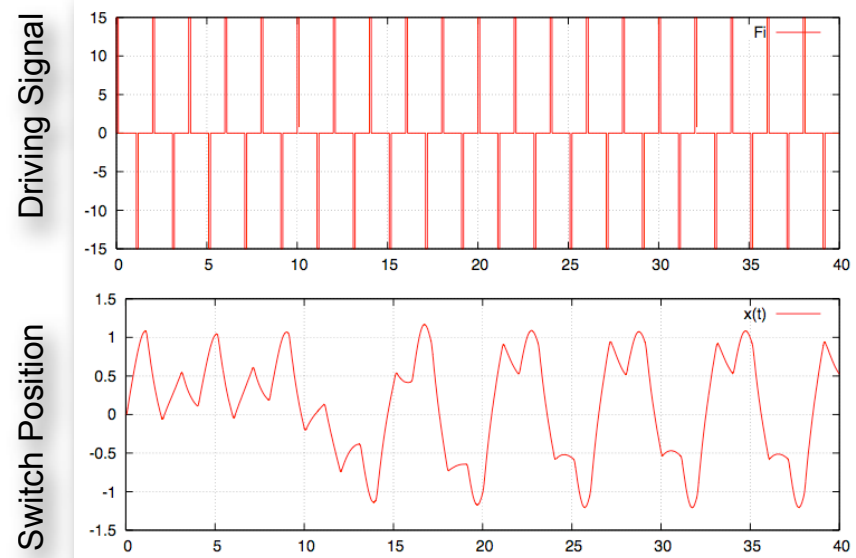- exhibits normal behavior and important potential failure modes

Example of a Properly Driven Switch:
A properly driven switch is in phase with the driving signal.
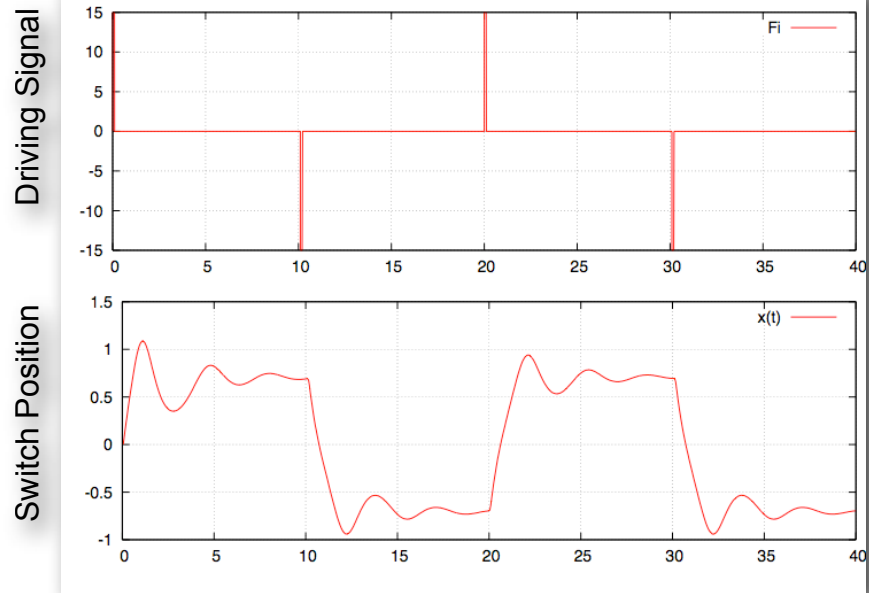
Example of an Improperly Driven Switch:
When driven at too high of a frequency or with too strong of a driving force, the switch's dynamics are unstable.

Integrated Simulation

A 1 kilobyte memory-bank example "circuit" of such switches was assembled as a standard Xyce input model and driven using a clocking source. From a reference, good design point, design parameters were assigned randomly at ±10% variations making each memory bit dynamically different. As these design space variations were increased, the previously working memory bank began to exhibit dynamic failures of non-switching and out of phase switching bits.

"We are actively working with 1700 to apply this technology to impact their products."

Scott Hutchinson, Manager Department 1437

Example of Normal Behavior:
Normal Behavior (several bits shown) illustrates variation in clocking over several bits in the integrated memory simulation.
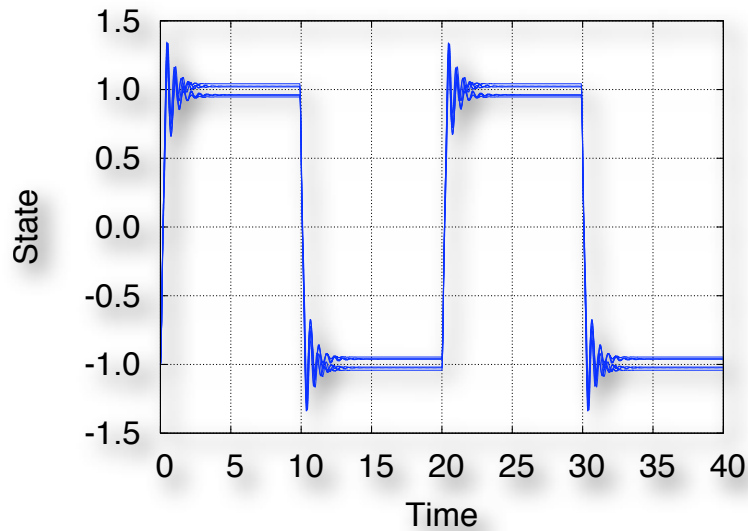
Example of Failing Behavior:
Increasing normal process variations causes a working memory bank to exhibit dynamic failures of non-switching and out of phase switching bits.
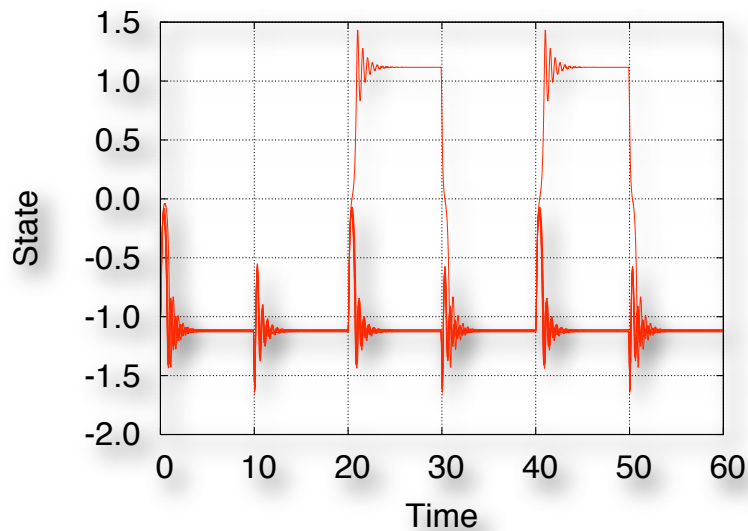
About Xyce™

The Xyce Parallel Electronic Simulator provides improved circuit simulation capability over other available tools in the following differentiating areas:

- *Size* - Capability to solve extremely large circuit problems at the transistor level by supporting large scale parallel computing platforms (up to thousands of processors). Note that this also includes support for most popular parallel and serial computers.
- *Speed & Robustness* - Improved performance for all numerical kernels (e.g., time integrator, linear solver) through utilization of Sandia's world-leading algorithms and solver libraries (http://software.sandia.gov/Trilinos) as well as novel techniques designed specific for Sandia's circuit modeling community
- *Flexibility* - Support for modeling circuit phenomena at a variety of abstraction levels (device, analog, digital and mixed-signal) in a rigorous and tightly coupled manner, allowing for timely, full-system solutions
- *Applicability* - Radiation-aware (x-ray, gamma and neutron) as well as aging device models. Additionally, most of Sandia's MDL technologies are supported (e.g., CMOS-6 and CMOS-7 with ViArray coming soon)
- *Portability* - A client-server or multi-tiered operating model, wherein the numerical kernel can operate distinct from the simulation interface (Xyce is part of the ESimTools project; please see http://sass2152.csu891.sandia.gov/esimtools for more information). Xyce also runs on most desktop systems including MS Windows (2000 and XP), Apple OS X, and Red Hat Linux
- *Maintainability* - Object-oriented code design and implementation using modern coding practices that ensure that the Xyce Parallel Electronic Simulator will be maintainable and extensible far into the future
- *Accuracy* - Rigorously verified and validated code and device-model implementations for regimes of interest to Sandia customers. This distinction is critical where design margins are tight and environments are extreme

Xyce is a parallel code in the most general sense of the phrase – a message passing parallel implementation – which allows it to run efficiently on the widest possible number of computing platforms. These include serial, shared-memory and distributed-memory parallel computers. Furthermore, careful attention has been paid to the specific nature of circuit simulation problems to ensure that optimal parallel efficiency is achieved even as the number of processors grows

As mentioned above, the Xyce Parallel Electronic Simulator has been developed in support of the unique requirements of electrical designers and analysts at Sandia and, as those needs continue to evolve, Xyce will adapt to continue aligning with those needs.

Trademark Information

Contacts

Email: mailto:xyce-support@sandia.gov
Web: http://www.cs.sandia.gov/Xyce
Dept. 1437: http://micro.sandia.gov